

Next generation lightweight mirror modeling software

William R. Arnold, Sr.*^a, Mathew Fitzgerald^b, Rubin Jaca Rosa^b, H. Philip Stahl^c

^aDefense Acquisition, Inc., Jacobs ESSSA Group, Huntsville, AL, USA 35806;

^bNASA Intern, MSFC, Huntsville, AL, USA 35812;

^cNASA ZP10, MSFC, Huntsville, AL, USA 35812

ABSTRACT

The advances in manufacturing techniques for lightweight mirrors, such as EXELSIS deep core low temperature fusion, Corning's continued improvements in the Frit bonding process and the ability to cast large complex designs, combined with water-jet and conventional diamond machining of glasses and ceramics has created the need for more efficient means of generating finite element models of these structures. Traditional methods of assembling 400,000 + element models can take weeks of effort, severely limiting the range of possible optimization variables. This paper will introduce model generation software developed under NASA sponsorship for the design of both terrestrial and space based mirrors. The software deals with any current mirror manufacturing technique, single substrates, multiple arrays of substrates, as well as the ability to merge submodels into a single large model. The modeler generates both mirror and suspension system elements, suspensions can be created either for each individual petal or the whole mirror. A typical model generation of 250,000 nodes and 450,000 elements only takes 3-5 minutes, much of that time being variable input time. The program can create input decks for ANSYS, ABAQUS and NASTRAN. An archive/retrieval system permits creation of complete trade studies, varying cell size, depth, and petal size, suspension geometry with the ability to recall a particular set of parameters and make small or large changes with ease.

The input decks created by the modeler are text files which can be modified by any text editor, all the shell thickness parameters and suspension spring rates are accessible and comments in deck identify which groups of elements are associated with these parameters. This again makes optimization easier. With ANSYS decks, the nodes representing support attachments are grouped into components; in ABAQUS these are SETS and in NASTRAN as GRIDPOINT SETS, this make integration of these models into large telescope or satellite models easier.

Keywords: Lightweight, mirrors, FEM models, modeling software, design trade studies.

1. INTRODUCTION

The design of lightweight mirrors for both space and terrestrial applications has evolved along with the wide variety of materials and construction methods now available. As the apertures grow and the wall thicknesses shrink in the never ending search for the optimal design, the importance of Finite Element Methods (FEM) in the design process becomes paramount. One of the constraining issues in the design/optimization process has been how long it takes to create the models. The complexity of the modeling effort can often result in 400,000 plus elements in even a "crude" model of an 8-meter class mirror. NASA has been participating in the Advance Mirror Technology Development (AMTD) program with industry and academic partners. The development of the Arnold Mirror Modeler has been encouraged and supported as part of that project.

Years ago, when faced with the need to optimize the design of the Kepler primary while working for then L3-Comm Brashear (now L3-Comm Integrated Optics Systems), I developed an automated model generator which made the creation of many alternative designs in a few hours rather than the traditional weeks per individual model. The challenge then as now was to design something which could be manufactured as well as launched. That predecessor program dealt with a single mirror substrate and used the development software of the day and ran on the Windows XP platform. When Microsoft launched it new Windows 7 platform, they unfortunately chose not to make their new 3D graphics package (XNA) backward compatible, so substantial portions of that original modeler could not be easily updated or expanded. Since Brashear owned the rights to the original software which was developed on their nickel, this new version was derived from first principals and a completely new approach to the grid generation problem. Indeed even the development language was updated to Visual Basic.Net 2010 (the closest thing to my beloved FORTRAN I could find).

The goal was a tool which could create models of any possible mirror construction method or configuration. As the current launch vehicle capabilities remain indeterminate, the need for the ability to compare segmented as well as monolithic lightweights became evident. With the continued evolution of ULE[®] based construction methods, the merging of segmented into monolithic structures was included in the modeler. The other materials have not been neglected either. By selection of options, all the construction methods appropriate for; pocket-milled Zerodur[®], cast Borosilicate and other materials can be modeled.

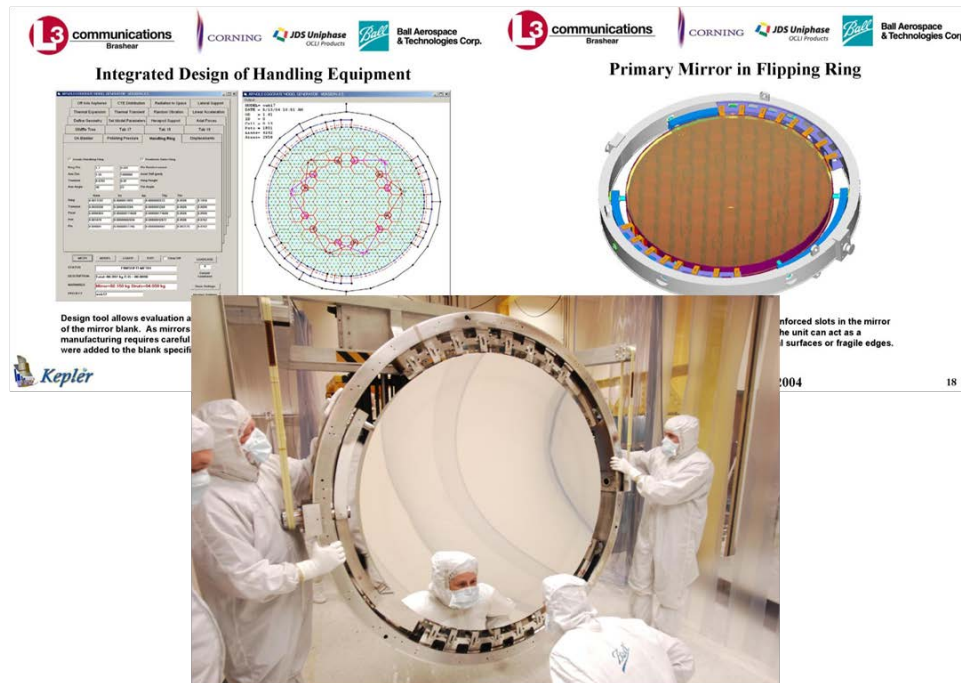


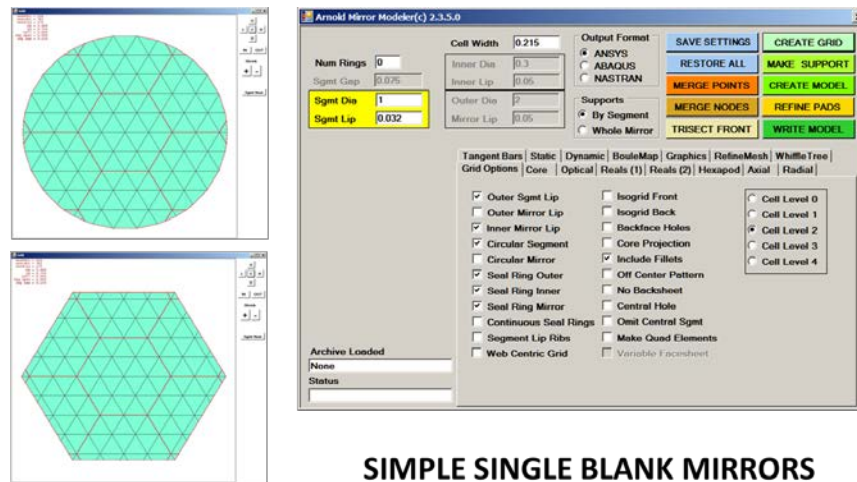
Figure 1. Predecessor modeler was used to design the Primary Mirror Assembly on the Kepler Planet Finder Mission. The above figure shows the progression from initial design trade study, complete assembly design and finally the handling equipment being used during the coating operations. Special reinforced zones were built into the mirror to interface with the ring and the ring's radial pads align with the internal ribs in the blank.

The current state of the user interface (UI) owns much of its ease of use and built in error checking to the tireless usage and stress testing of my co-authors Matt and Rubin. With only limited documentation and training they were given specific modeling tasks, which they accomplished with excellent results. Many of the features incorporated were direct results of problems encountered during executing their trade studies, and their diligence in finding and documenting the numerous bugs and inconsistencies.

2. ARNOLD MIRROR MODELER CAPABILITIES

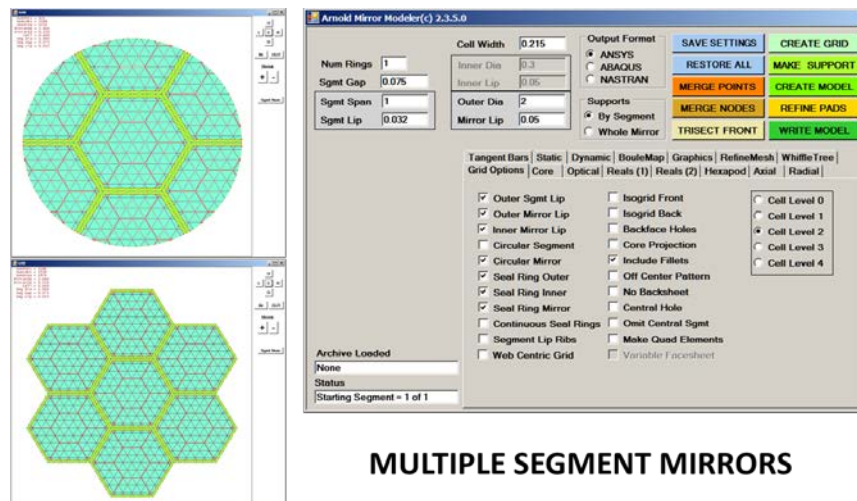
Enough history, let's talk about what the program can do and how fast it can do it. Whether it takes seconds or minutes to create a given model depends mostly upon the computer being used. On a powerful analysis or gaming computer, a 400,000 element model can be generated in 2-3 minutes, on a simple laptop with minimal graphics capabilities, that same model can take 5-6 minutes to generate (and display). Whatever machine it runs on, it will cut weeks off the typical design trade study. The speed will allow aggressive investigation of geometric (cell size, depth, etc.) as well as the usual wall thickness and support stiffness variations. One of the most powerful uses is the in the design of suspension systems integral with the substrate. Special features were added to make it easy to see how the support fits the cell pattern, even on very large models (the grid is generated only once, successive trial suspension locations can be plotted until satisfactory pattern found, then the model is generated).

The following figures will show the majority of the available options. The "CREATE GRID" button generates the grid pattern based upon current settings and displays the result. The pictures will do most of the talking for this stage.



SIMPLE SINGLE BLANK MIRRORS

Figure 2. The UI controls all aspects of the model being created. When “Num Rings” is zero, a single mirror segment is generated. The key variables are in yellow box. The options tab exposes configuration choices. Mirror shape can be circular or hexagonal.



MULTIPLE SEGMENT MIRRORS

Figure 3. The user simply changing the “num rings” variable switches the UI into multi-segment mode. The “circular mirror option controls the outer shape between complete hexagonal segments and trimmed to a circular aperture.

There is currently an upper limit of annular rings, set at 5 concentric rings. This value can be extended if the requirement ever arises. The segment pattern can be hexagon centric or intersection centric, see figure 4.

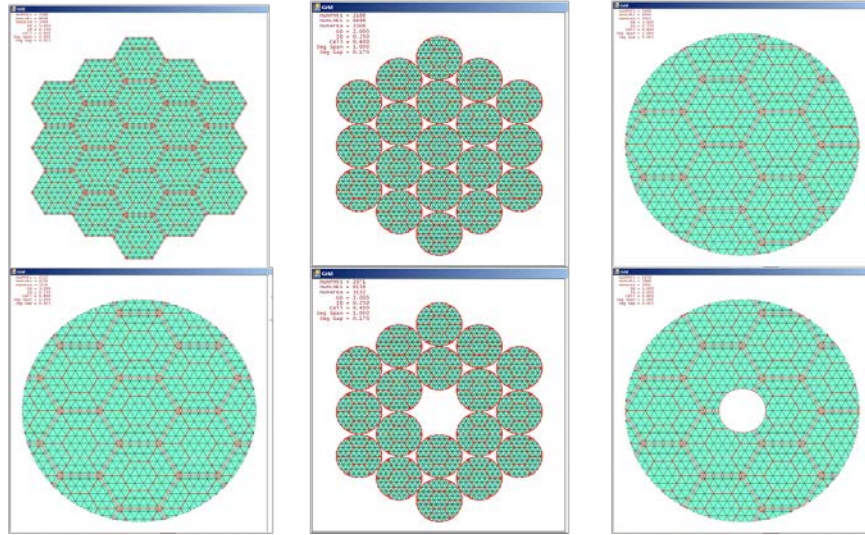


Figure 4. The potential variations are almost limitless. The segments do not have to only be hexagonal.

The next figure actually highlights two capabilities, merging segmented grids into a unified grid and grid display options. In the merge points process the joint details are preserved. The other capability is the pan, zoom and element shrink features. This helps the user understand the more complex grids and identify potential badly formed elements (aspect ratio). The point merge feature can be used to eliminate many but not all of these. Any automated mesh will have the potential for these problems whenever the regular pattern is cut by a boundary. Cell size and super-cell (segment) ratios can create poorly formed features, the ease of adjusting small values and redisplaying the grid is a major advantage.

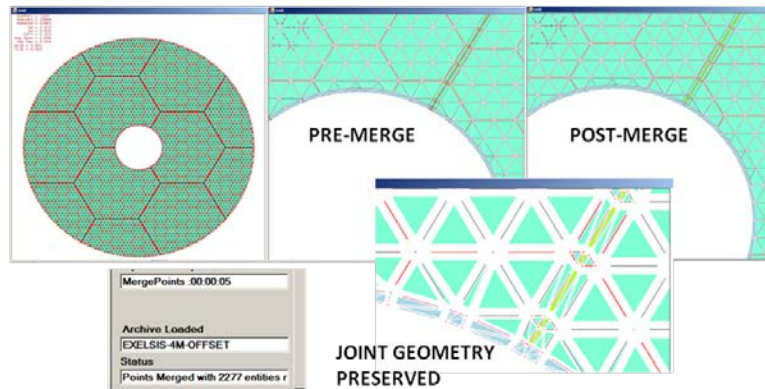


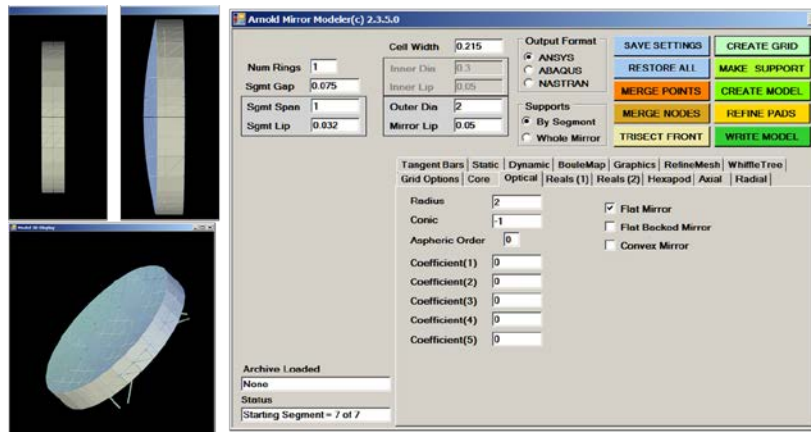
Figure 5. After the merge operation completes, it replots the grid. The user can evaluate the affect and the UI gives the statistics on points merged and the time elapsed for that function.

When dealing with large models, some of the operations take a noticeable amount of time, to let the user know something is happening and the program is not hung up, during these longer operations various counters and progress bars are displayed. Once the user is satisfied with the grid, he can display the 3D model by pushing the “CREATE MODEL” button.

We will get to suspension systems later. Figure 6 shows some of the options available for the core. To support AMSD style mirror (glass on composite), the core can have non-glass materials. The thickness and material can be controlled by layer (up to current 9 layers). Once the model is created and displayed, the modeler give critical statistics such as size of model, weight and aerial density. During initial iterations you do not even need to run the FEM model to get a reasonable weight estimate. These statistics now include information on how much of the mass is attributed to the faceplates and core, as faceplate light-weighting is becoming a critical design consideration as size increases.



Figure 6. Core options panel. The layer thickness sub-panel will only appear if the multi-web-thickness option is selected or if non-glass core options are activated.



7. Complete control of the optical prescription is available. In addition flats, flat backed concave and convex mirrors, and completely convex designs can be generated.

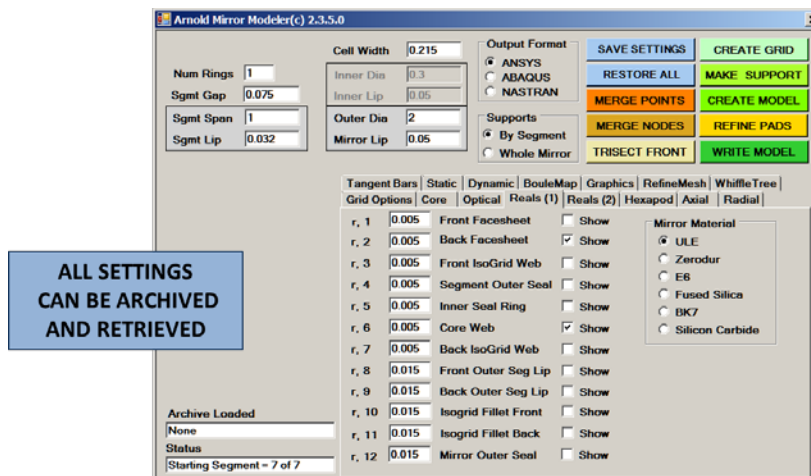


Figure 8. The Reals panel exposes most of the mirror variables and materials.

Every variable in the generated FEM model can be preset in the UI. Both the UI and the resulting input files to the supported FEM programs are fully commented, so the user can easily alter these values and understand what features are represented. Remember the little “show” checkboxes; we will explain their use later. Since trade studies are the main thrust of this program, special archiving features preserve a complete settings environment. By use of these functions, the user can return to previous models, make minor (or major) alterations and save under a different name.

2.1 Working with suspension systems

Now we can get to the various suspension systems possible. As the system can model arrays of mirrors, suspensions can be applied (identical geometry) for each segment or to a whole mirror if segments merged. There are some limitations, pattern generators for segmented mirror “trimmed” to a circular aperture do not have the artificial intelligence yet to do supports for these odd shapes (working on it, but definitely back burner stuff!), so for now if you trim the segments, you can only use the whole mirror option.

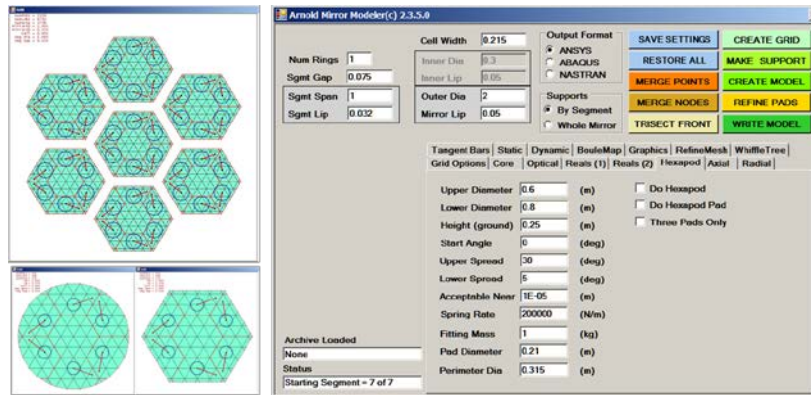


Figure 9. Hexapods are one of the most useful kinematic supports. The UI gives you complete control of the geometry and clocking relative to the grid. There are two popular variants of this, three pads on the mirror and six pads.

Axial supports have been long been useful for polishing mount and test mount designs, and more recently for auxiliary launch support systems when used in conjunction with hexapod or other kinematic operational supports.

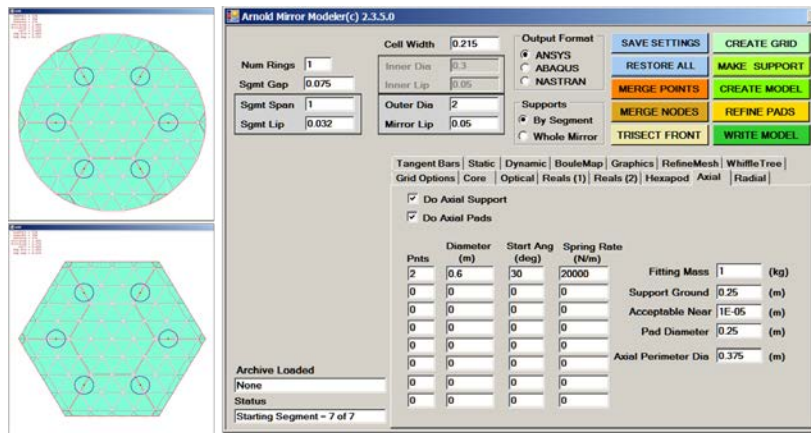


Figure 10. The axial mounts are defined in concentric rings, with number of points, clocking and separate radius for each ring.

The radial support for right now only allows a simple equally spaced single row. Eventually Schwesinger style lateral supports will be included (both uniform spacing and uniform force variants). For now, the choice is radial with tangent bar or not. Again the radial support in conjunction with an operational kinematic system can be used to model an auxiliary launch support.

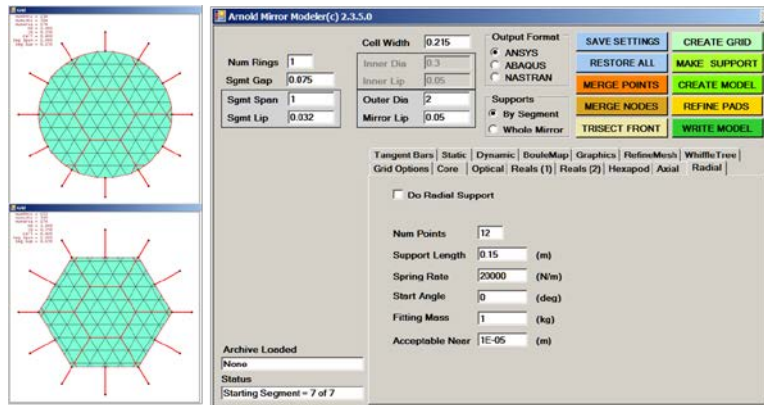


Figure 11. Not much more to say about this one, it works with hexagon or circular designs.

The other traditional system is the tangent bars; the user can use one, two or three bars. For now the spacing is set, 180 degrees for two and 120 degrees for three bars.

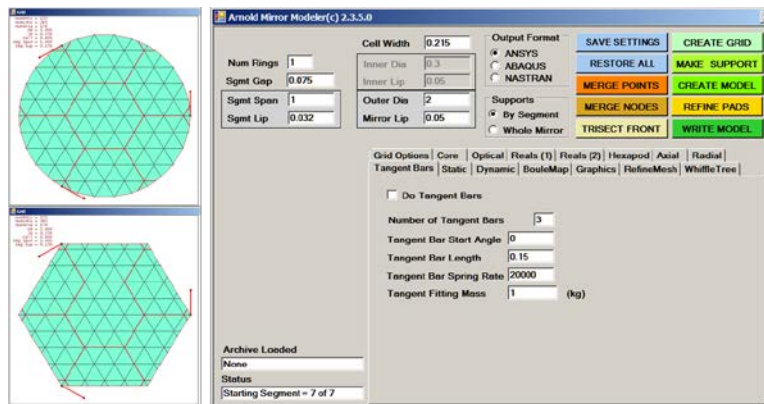


Figure 12. Again, there is not much more to say about this. (Note: this UI shot did not make these plots, just shows panel)

Whiffle tree supports are mostly used for smaller mirrors, but I am trying to build a more complex capability, but the multi-layer whiffle trees have an awful lot of combinations. Creating default settings for each level has not proved easy.

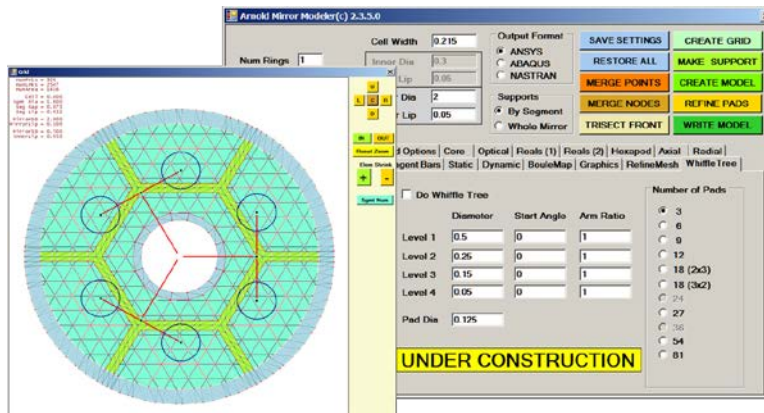


Figure 13. Here is a simple example of a whiffle tree support, not too earth shaking for simple combinations.

Now we can look at more of the trade study related features. The program generates text files which are inputs to the major FEM programs. While some may have other choices, ANSYS, ABAQUS and NASTRAN dominate the market. During most trade studies, you use the same loading conditions for each run. The program lets you preset these quickly, and when combined with the archive system, you can generate “canned” studies with ease. As ANSYS has a full

scripting language APDL, the program writes a complete analysis, with plots, result files for further processing, etc. ABAQUS input will run the complete analysis, but output plots and files are usually done from their graphical interface ABAQUS/CAE. Eventual I will write some Python scripts to automate the plotting like the ANSYS decks currently do. NASTRAN is well a little behind the others graphically. Internally, it still makes CALCOMP plotter files (for those with working 70-80's plotters ... Great!). Most NASTRAN users will use FEMAP or PATRAN to make pretty pictures.

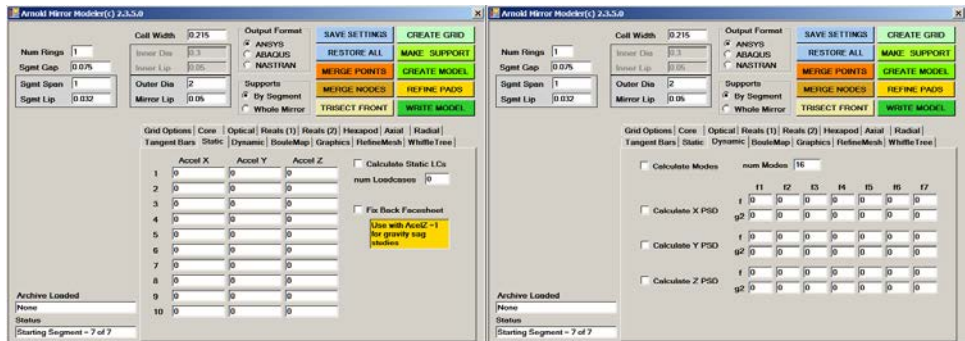


Figure 14. This figure shows the Static panel and the Dynamic panel. Both can be used at the same time, the program will always do the static runs first, then the modal dynamics, then PSD cases (ANSYS will mess up otherwise).

If you are modeling a ULE mirror, and cryogenic performance is critical, or high accuracy at operational temperatures is important, then the following capability will be useful. If you have the data (the samples shown were potential boules for use in the Subaru 8 meter) then the task of mapping that data into even a modest size model is non-trivial. A complete input, editing and assigning each boule to locations in the mirror is provided. Once the data is entered, the archiving system can be used to evaluate using each boule in various locations. [Note: no claim is made that this process is fast, have lots of coffee ready, but it will be hours at most, not weeks]

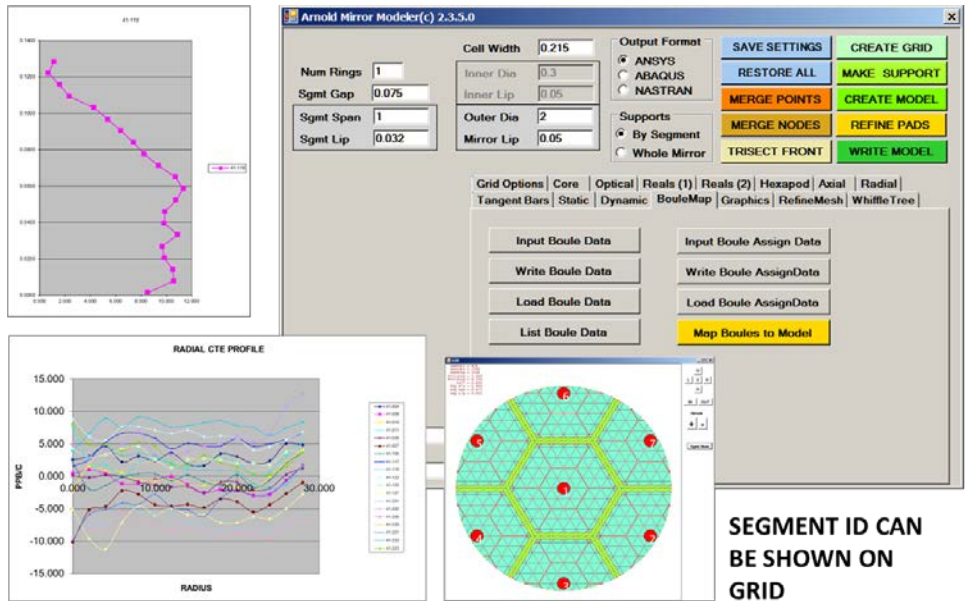


Figure 15. The boule mapping panel has routines to read in raw data on each boule, then assign all or portions of that boule to be used as core, face plate etc. For each element node the position in boule is determined, the CTE calculated, then averaged for that element. The average CTE is then located into a predetermined set of CTE ranges (separate material for each range, up to 100 increments).

Now we can get into the user friendly features. Most of these were added one we started into the large merged grids. The time required to regenerate the grid started to get excessive, and once displayed it was almost impossible to understand what was really happening. The simplified mesh display came first, then the pan and zoom. It still was hard to see the fine details and also to verify that elements were actually generated where you wanted. Then the element

shrink was implemented. Right now the buttons in the grid window work and alternately the numerical keypad can be used. Mouse only manipulation currently only works for zoom.

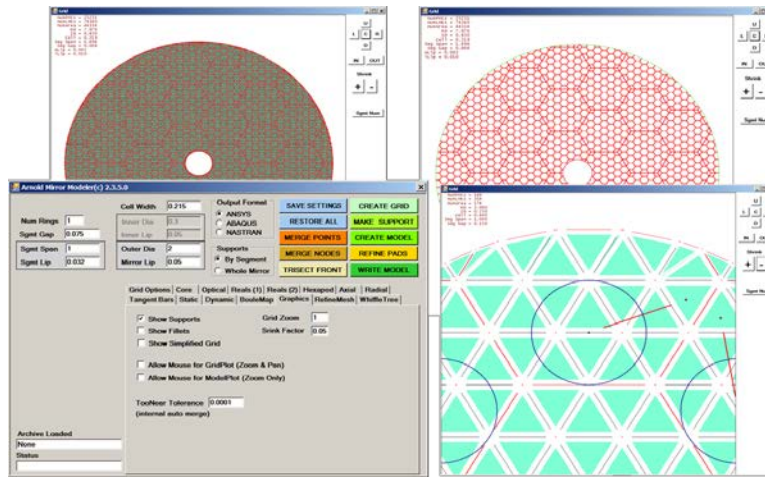


Figure 16. This figure highlights the main user graphics convenience features.

The next feature is particularly important with the more fragile designs. There are several options on attaching the suspension components to the mirror. The simplest method is to find the nearest mirror node to the end of the support element and CERIG (rigid body or RBE3 or constraint equation) the two together. This puts concentrated point loads into the mirror, so it is not recommended. The next level is to identify all the mirror nodes (on back face) inside of the “pad” diameter and CERIG them all to the master node (end of support element). This still does not properly load the mirror, but is a little better. The final option is to refine the mesh under the pad. This option actually increases the mesh under the pad. All elements which touch the pad diameter are split; the result is a reasonable approximation to the pad edge. Now all nodes on edge or under the pad are CERIG’ed to the master node. This is still not perfect, but the pad edge is usually the critical zone and you at least have good approximation there.

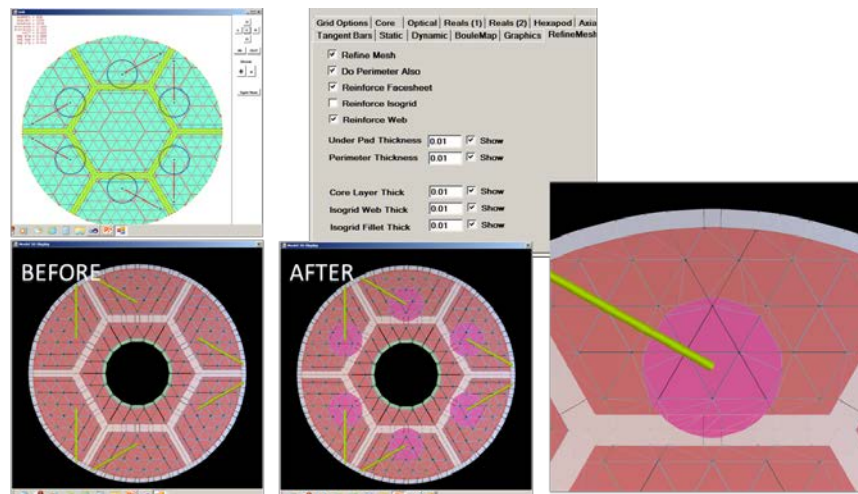


Figure 17. This figure shows a typical mesh before and after the REFINE PADS button pushed.

The next figure shows what more combinations of the options can be used to produce. While the face sheets front and back are always triangles, the core and isogrid elements can be either triangular or quadrilateral. The back face sheet can have pockets or eliminated altogether. If optical performance predictions for gravity sag or higher frequencies are planned, the front face sheet can be refined without increasing the core or back face sheet mesh. Pocket milling the face sheets prior to assembly has become increasing common. For very lightweight designs, the fillets from core and isogrid pockets represent substantial weight and can be included in the model.

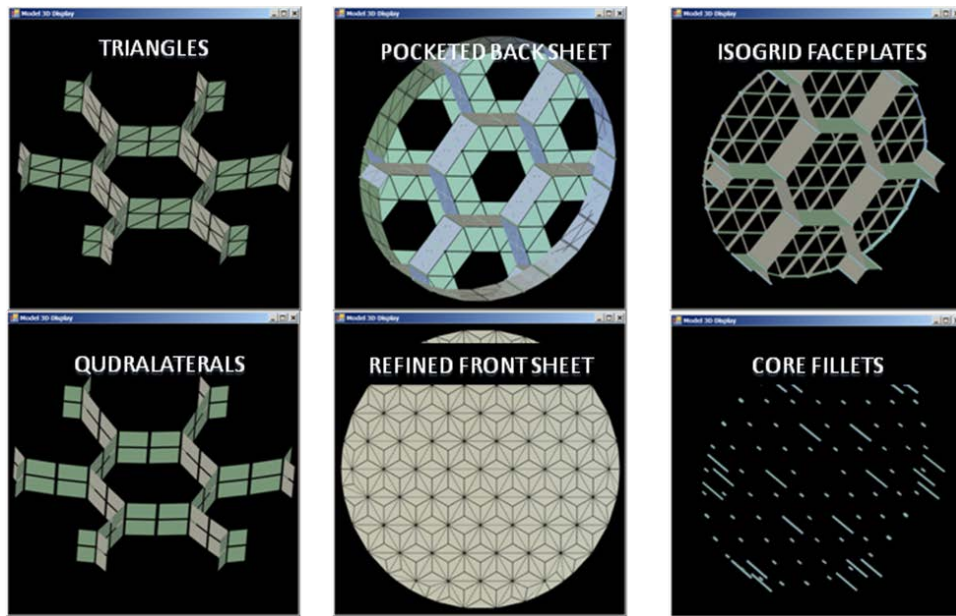


Figure 18. Various model options illustrated for simple model.

So, how is all this magic done? The key is the hidden back in figure 3. Over on the right side of the options panel is a group of radio buttons (means only one selection allowed from the group) labeled “cell level #”. Each level is a complexity level of the basic grid pattern. Level 0 basically makes an isogrid core, not much else; you can have seal plates, hexagonal or circular, multi-segment etc. But they will all be isogrid.

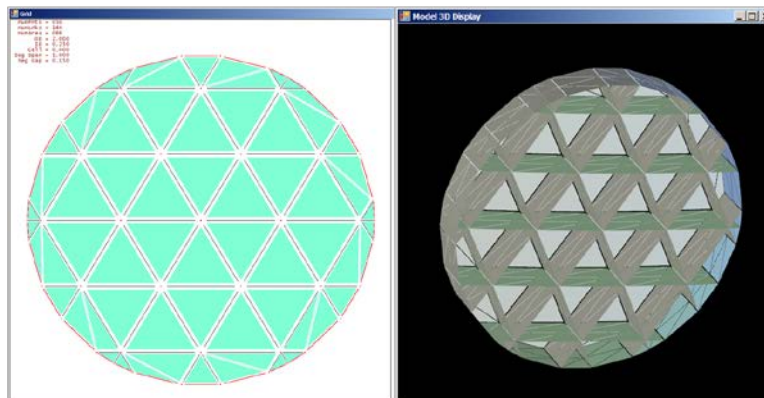


Figure 19. This is about the simplest pattern possible.

Level 1 can generate hexagon cell with or without isogrid faceplates front and/or back, with 6 sub-cells each.

Level 2 can generate hexagon cell with or without isogrid faceplates front and/or back, with 24 sub-cells each.

Level 3 is actually a variant of Level 2, the main difference is the front isogrid has 24 cells and the back has 6 cells.

Level 4 has twice the element density of Level 2 or Level 3. It is mainly intended for higher frequency work or checking you mesh quality. Using level 4 on large models is not recommended, but will work if you have the time and disk space.

The following figures show the various combinations possible for each level.

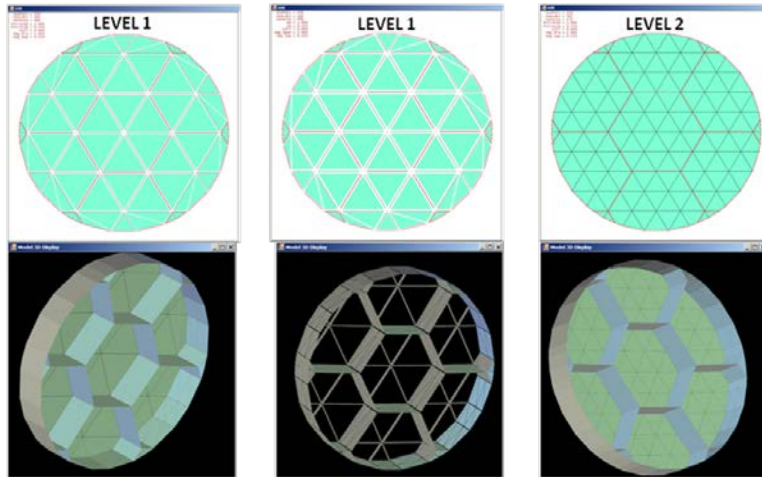


Figure 20. On the left you have a simple hex cell pattern, the center has the simplest of the isogrid faceplates, the right has a simple hex cell, but the faceplate has 24 elements for gravity sag studies. You can Trisect Front for even better resolution.

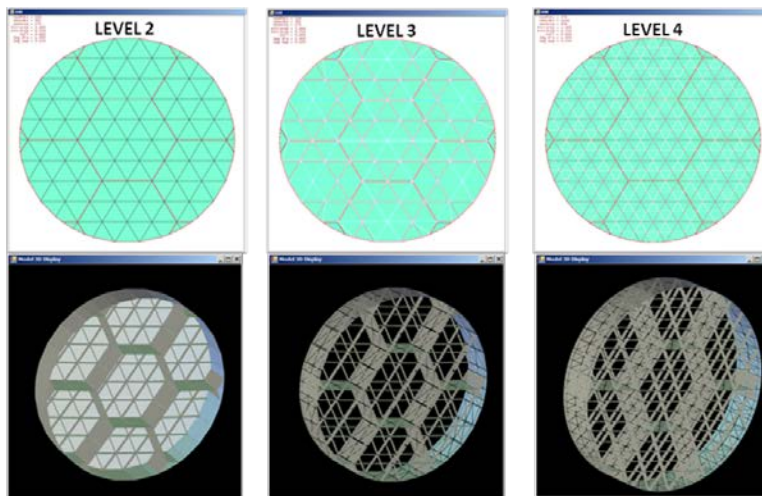


Figure 21. The left shows a 24 pocket isogrid, the center shown 24 pockets in front with 6 pockets in back; the right shows the higher density model.

The 3D display of the model can be panned and zoomed using the numerical keypad. The real constants can be selectively displayed or hidden to improve understanding of the model. I am still experimenting with appropriate color combinations for each type of feature.

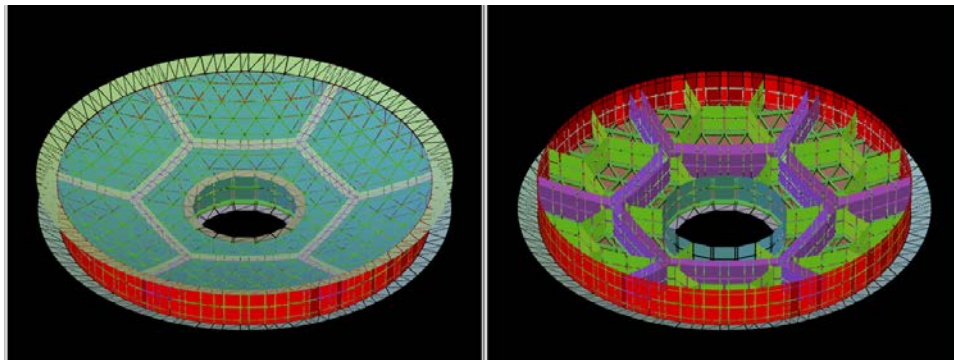


Figure 22. The 3D display can be rotated about two axes, panned up or down, left or right as well as zoomed in or out.

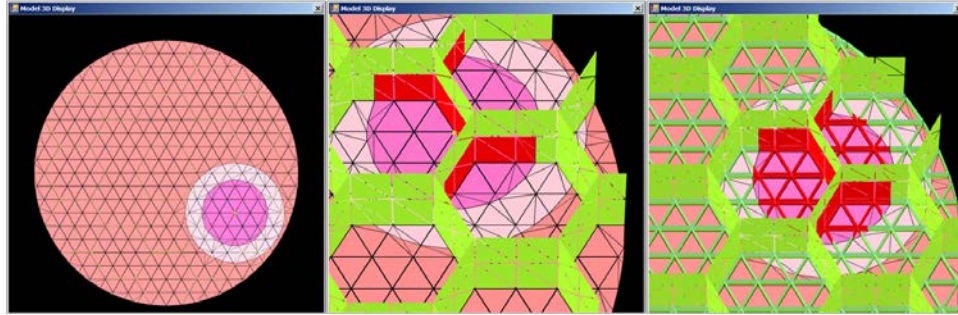


Figure 23. Localized mesh refinement and core reinforcement under pads. Also works with isogrid back plates.

3. STATUS AND WHERE WE ARE GOING NEXT

- Currently the program is undergoing NASA ITAR review to determine if any distribution restrictions are applicable to the program.
- NASA is working on distribution, licensing, revision control and error reporting mechanisms.
- A User Manual and tutorials are under development.
- A short course or seminar is under discussion for use of the program and mirror design methods.
- List of possible enhancements and requested features is growing daily.

ACKNOWLEDGEMENTS

This work was performed under the JACOBS ESSSA contract NNM12AA41C at NASA MSFC in support of the Advanced Mirror Technology Development project which is funded by a NASA ROSES SAT (10-SAT10-0048). We would also like to thank the NASA Graduate Student Researcher Program and the NASA Intern Program for its support of this project.

REFERENCES

- [1] Arnold, W. R., "Kepler Photometer Primary Mirror Assembly Current Status," NASA Tech Days (2004).
- [2] Arnold, W. R., et al, "Integration of mirror design with suspension system using NASA's new mirror modeling software," Proc. SPIE, 8836-17 (2013).